CREATION OF A COMPUTATIONAL FLUID DYNAMICS MODEL USING A BUILDING INFORMATION MODEL FOR VISUALIZING INDOOR ENVIRONMENT

A Thesis

by

YUNKUN CAI

Submitted to the Office of Graduate and Professional Studies of
Texas A&M University
in partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE

Chair of Committee,	Julian Kang
Committee Members,	Wei Yan
	John Bryant
Head of Department,	Joe Horlen

May 2016

Major Subject: Construction Management

ABSTRACT

A building's indoor environment has a significant impact on people's work efficiency and quality of life. The amount of energy consumed to control the indoor environment comprises a large portion of maintenance costs. Understanding how efficiently we can control the indoor environment, therefore, has been one of the critical issues in building design. Computational Fluid Dynamics (CFD) is one of the techniques that industry professionals use to understand fluid flow, heat transfer, and indoor chemicals movement, which eventually enables them to analyze indoor environmental conditions.

However, the complex process of CFD data modeling and the time-consuming CFD simulation process have been hindering construction practitioners from using this technique. As a data-rich geometric model, Building Information Modeling (BIM) may provide them with an alternative solution to create a CFD data model.

This study tests if it is possible to produce a CFD model using BIM. It also examines any challenges one may need to deal with in the process of creating the CFD model. For this test, a CFD model of Francis Hall, one of the old buildings on the Texas A&M University campus in College Station, was created using the Building Information Model that the general contractor used for building renovation. This research work also presents problems and barriers experienced in the course of the BIM-based CFD modeling process. The significance of this study is that it (1) verified the feasibility of recycling the Building Information Model and expanding its application areas and (2)

provided information for future software developers to improve the interoperability of CFD software and BIM software.

DEDICATION

To my family, my committee and my friends whose unwavering support brought me to succeed.

ACKNOWLEDGEMENTS

First of all, I want to thank my family for their emotional and financial support, without them, I would not have a chance to spend two years in this rich and competitive program provided by Department of Construction Science at Texas A&M University.

Then, I want to place my sincere gratitude to my committee chart Dr. Julian Kang, for his inspiring and valuable guidance through the entire research. I also wish to thank my committee members Dr. John A. Bryant and Dr. Wei Yan for their patient instructions and helpful suggestions.

Last but not the least; I would like to thank all the Creative Construction Lab (CCL) group members Jaeheum, Bardia, Javier, Sachin, Reza etc., for those who encouraged and helped me when I went though some hard times.

NOMENCLATURE

AEC	Architecture Engineering Construction
AHU	Air Handling Unit
BCA	Certification Standard for New Building
BIM	Building Information Modeling
BPS	Building Performance Simulation
CFD	Computational Fluid Dynamics
DNS	Direct Numerical Simulation
ES	Energy Simulation
HAI	Healthcare Associated Infections
IFC	Industry Foundation Class
IR	Infrared
LES	Large Eddy Simulation
RANS	Reynolds Averaged Navier-Stokes

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER I

BACKGROUND AND INTRODUCTION

1.1 Indoor Environment in AEC

Advanced technologies and new applications are increasingly utilized in the Architecture Engineering Construction (AEC) industry. Changes and upgrading in materials, structures and appearances are applicable from early design to post occupancy operation. However, an essential function of buildings is to provide healthy and comfortable shelters for humans. One of the most important concerns for construction practitioners is the indoor climate of buildings. The indoor climate can be categorized into four main factors: sound, light, indoor air quality and thermal climate (Cehlin 2006). Among them, the indoor thermal climate is a crucial and complicated part to study. Air temperatures, ventilation, air flow patterns, pollutant concentrations and air velocities are important factors that would have a significant impact on indoor thermal climate (Wargocki 1998).

Controlling indoor thermal climate has important values for different types of buildings. For business buildings, the indoor climate directly related to people's working efficiency. A comfortable indoor environment requires appropriate temperature, well-proportioned air distribution and ventilation. For healthcare buildings, healthcare-associated infections (HAIs) are regarded as severe challenges. Controlling air quality and airflow distribution is one effective measure to prevent infections caused by airborne microbial contaminates (Autodesk 2012). For laboratory space or data centers, temperature and ventilation control guarantee that equipment will operate properly without external disturbance. Another important issue for indoor climate control is saving energy. Every year, nearly half (47.6%) of all energy produced in the U.S. is consumed in the building sector. Of the electricity consumed, three-quarters (74.9%) goes to operate buildings which people live and work in (Architecture2030 2013). Proper operation of HVAC systems and other facilities to control indoor climate will make a great contribution to reducing energy consumption and post-occupancy operation cost.

1.2 CFD in Construction

It is not easy for designers and facility managers to study indoor thermal climate. The invisibility of air and temperatures make them difficult to trace and monitor. Without a direct view of indoor environment, it is difficult for building designers and mechanical engineers to optimize their design especially during the schematic design phase. Besides, due to the physical property of the air, mapping out indoor climate requires people considering velocity, temperature, high spatial and temporal variability of air. The essential problem for studies aiming at indoor climate analysis is a lack of effective approaches and techniques to monitor, visualize and analyze indoor factors like airflow, temperature and velocity. Previous studies relied on single-point techniques, such as thermocouple, thermistor, hot-wire anemometer, laser Doppler velocimetry and passive gas tracers that could only detect airflow and temperature where sensors were placed, but large-scale detection is troublesome and inefficient (Cehlin 2006).

From the end of the 1990s, studies started to focus on using new applications and powerful simulation modeling tools to visualize complicated activities of indoor climate. Computational Fluid Dynamics (CFD) is a very powerful numerical method that has been applied to Architecture Engineering Construction (AEC), Petroleum Engineering, Meteorology and many other industries. In the ACE industry, CFD has been mainly utilized in Building Performance Simulation (BPS) to study issues like wind environment, heat transfer, pollutant dispersion, indoor environment improvement and HVAC system design. Compared to traditional methods like the wind tunnel for outdoor airflow analysis and single-point sensors for indoor environment study, CFD has shown its great power for its convenience, economy and accuracy.

Usually, there are three main steps to conduct a CFD simulation in currently used software: pre-processor, flow solver and post-processor (Ranade 2001). The CFD process begins with defining the target geometry, generating the grid and inputting the boundary condition and flow pattern to these grids. Then, the solver phase is the critical step to run the equations calculation based on the flow conditions provided. Last, the post-processor produces the visualization to present the outcomes. When using CFD in building analysis, the first thing is to build the geometric model. But the time and efforts cost on this modeling process would be a barrier that hinders building designers from using this tool. One way to solve this problem is Building Information Modeling (BIM).

1.3 BIM and CFD

Building Information Modeling (BIM) is an innovative technology which is being utilized in an increasing numbers of projects. The US National BIM Standard (2007) defines BIM as a digital representation of physical and functional characteristics of facility. BIM is not limited to a 3D model. It involves in the whole lifecycle of a project; it helps in design innovation, promoting informed decision making during construction and proactive decision making in facility management. The interoperability of BIM allows it to carry data in other ways and makes it possible to expand BIM to a broad area, such as lighting, wind, energy or other analyses.

CFD is another emerging area that can integrate with BIM. The building information model is meant to contain all the geometry information like site characteristics, building components location and relation and operational parameters in one complete digital model. The simulation capabilities of BIM applications support the use of CFD that can provide insights into critical indoor environmental factors, such as airflow patterns, temperature detect and thermal comfort (Autodesk 2012). Incorporating BIM with CFD would make the airflow and thermal condition simulation smarter by taking advantages of the information that is already embedded in the building geometry model. This may simplify the traditional CFD simulation process and extend its usability in AEC industry.

1.4 Case Information: Francis Hall and Chilled Beam System

Built in 1918 and used for veterinary study, Francis Hall is situated in the middle of the campus of Texas A&M University. The structure remains a classically proportioned three–story reinforced concrete building with brick and cast stone exterior. When it became vacant, Department of Construction Science recognized a good opportunity to provide a high-quality learning environment and to expand student enrollment. The innovation process was full of challenges. For instance, the innovation in auditorium involved some structural changes; the nearly 100-year-old building had to comply with modern building codes; the new interior also featured exposed HVAC system for students to understand how they operate. The innovative building has many special features, including specialty labs for estimating and surveying, building information modeling facility, a video conferencing room and an exhibit hall.

In order to create a better indoor environment, an active chilled beam system is used in this building. A chilled beam is a type of convection HVAC (heating, ventilation, and air conditioning) system designed to provided cooling only for large buildings (Oughton and Hodkinson 2008). This system is predominantly used for cooling and ventilating spaces, where a good indoor environment and individual space control is valued.

The chilled beam in this building is an active chilled beam, which has 2-way supply air discharge and one single piece return air grille. In the air return vent, there is a 2-row horizontal cooling coil filled with chilled water. The primary air is supplied by a 100% dedicate outside air system and this fresh and dehumidified air is transferred to the chilled beams via ducts; the secondary room air is induced directly from the conditioned space through the cooling coil. They are mixed to condition the room.

The air circulation path in this building is started from the air handling unit (AHU). The center air handling unit dehumidifies fresh outdoor air and brings it into the building. Then, the treated fresh air is circulated through the duct system and distributed to each chilled beam. Different from other regular air handling unit, it doesn't have any ducts for air return purpose. The air is expelled through two exhausted fans which are on the top of the roof. By understanding the basic information about the chilled beam system and air traveling path, this study will set data for CFD analysis based on the actual case. The mechanical knowledge is also helpful to test the simulation results roughly.

On the one hand, Francis Hall has an exposed HVAC system and the indoor thermal environment has never been investigated before, on the other hand, the building information model provided by general contractor gives a perfect opportunity to integrate with CFD simulation. These are the incentives to choose this building as a study case.

CHAPTER II

PROBLEM STATEMENT AND RESEARCH OBJECTIVE

2.1 Problem Statement

Although previous studies have established a firm foundation of using BIM and CFD to study building indoor climate or wind environment, problems and gaps still exist while improvement and progress need to be put forward.

Firstly, studies have noticed that the CFD could be a powerful tool for construction industry. BIM can provide many data needed for the initial building geometry modeling of CFD. Most of those studies were focused on integrating BIM and CFD by building frameworks and using intermediate files or tools. However, most BIM applications are capable of transporting models to CFD tools. This may be the most straightforward way to combine these two tools. But few researches have paid attention to using existed building information model for the CFD simulation process.

Secondly, the target size and range of previous studies were mainly limited to a single room or a small area of buildings, no matter the traditional CFD or the chilled beam study cases. The time-consuming modeling process could be one of the reasons. Since most current projects have already adopted BIM as a common technique, which makes large geometry models are available for CFD simulation. Using CFD for large scale building simulation is approachable. There is a need to investigate the CFD simulation for large building area.

Based on the gaps of previous studies, this research is going to determine:

- If the building information model can be applied for CFD simulation.
- If yes, what is the basic workflow of combining these two tools and how long this process is going to take?
- If not, what are the problems and barriers that impede in this process?

2.2 Research Objective

Based on the problems mentioned above, these are the main objectives in this study:

- This study is going to utilize the Francis Hall building information model to conduct CFD simulation and visualize the airflow distribution and temperature variation of the chilled beam system.
- This study is going to streamline the basic workflow for applying building information model to CFD simulation analysis.
- This study is going to record the time consumption of each stage in the workflow.
- This study is going to determine what are the problems and barriers during the integration process. How can they be improved?

2.3 Research Case and Tools

Francis Hall was used as the only case in this study. The basic information of this building was introduced in the first chapter. There are several reasons for choosing this building: 1) as a recently renovated building, inner structure and MEP equipment are exposed intending to help students have a clear view. The basic ventilation mechanism is known. So it can be used to test the simulation results to some degree; 2) Francis Hall is the first building in Texas A&M University which adopts chilled beam system as the HVAC equipment. The indoor environment has never been studied and tested before. It is a good opportunity to use CFD to simulate and visualize the performance of this system; 3) the building information model of this building is already provided by the general contractor. There is no need to spend time on creating the model.

Autodesk Revit is the BIM software used in this study. So far, it is still the most prevalent BIM tool in construction industry. The original building information model provided by general contractor is in a Revit file. Besides, the free access of it gives the author a better understanding of it.

Considering the interoperability of BIM and CFD tools, Autodesk Simulation CFD is used as the CFD software in this study. It is a CFD and thermal simulation tool with a wider applicability. This software is helpful to conduct an overall CFD analysis for building indoor environment.

CHAPTER III

LITERATURE REVIEW

This chapter introduces many important studies that related to this study topic. Four sections focused on the four core elements of this research which are indoor environment, CFD principles, chilled beam system and BIM.

3.1 Mathematical and Experimental Approaches

Different models and approaches are put forward to provide people an insight to monitor, analyze and control indoor climate. For example, Schauberger et al. (2000) designed a steady-state model to calculate indoor heat transfer, odor mass flow and indoor ventilation of a livestock farmhouse. This model was applied to a prognostic mode to assess the indoor climate including thermal parameters and air quality. Equations and parameter calculations were the essential steps used in this study. Finally, the result and comparison were presented by charts and tables. And the model has been verified for a wide range of input parameters (Schauberger et al. 2000).

In other studies, equipment like sensors and infrared cameras are used to monitor and trace indoor air and optimize design to acquire comprehensive data and information. In Ivanov et al. (2002)'s study, a distributed sensor system was developed in a smart-home lab to monitor the indoor climate including air humidity and velocity, temperature and CO_2 concentration. In this experiment, three prototypes of technical solutions were assigned: multi-gas sensors module were used to detect air composition, temperature, dust and humidity and collect data; wearable wireless devices were designed to measure surroundings of users. Then, data loggers helped to save these data through BGA micro-sensors to integrate these components to a sensor network(Ivanov et al. 2002). This study presented a comprehensive continuous monitoring system to detect indoor air quality and pollutant concentration.

In addition, scientific visualization provides people an explicit way to see the actual indoor climate by transforming raw data to comprehensible image. It facilitates the process of identifying problems and reduces confusing details (Cehlin 2006). A study aiming at indoor climate and power usage of a data center utilized infrared (IR) cameras to visualize the air flow pattern and air temperature. The image can help researchers to visualize the improper usage of chilled air and help them to check parameter calculation (Karlsson and Moshfegh 2005).

3.2 Numerical Simulations—Computational Fluid Dynamics

Methods of studying indoor climate are not only limited in data analyses and experimental approaches, numerical simulations have shown its power in more and more aspects. This new solution avoids the effort and cost spent on the whole-scale measurement of experiment approaches. It allows people to visualize the actual indoor climate by simulating the study cases. Among these simulations, Computational Fluid Dynamic (CFD) is an attractive and powerful technique to achieve climate simulation and visualization. It can be applied to show indoor airflow patterns, temperatures, heat transfer and some other factors to help to analyze indoor thermal climate (Sreshthaputra et al. 2004). Another great contribution of CFD is to optimize indoor environment design and reduce initial cost and operation cost. It makes indoor mechanical system function efficiently while reducing energy consumption.

CFD makes use of a fundamental set of equations to describe the essence of fluid flow. These equations come from three basic principles: conservation of momentum, conservation of mass and conservation of energy within the fluid (Den Hartog et al. 2000). Although these equations remain the same when analyzing indoor heat transfer and air velocities, as they are used for different parts of a building, the different boundary conditions change. This makes it impossible to use analytical solutions for indoor climate modeling. Computer-based iteration of these equations is used to achieve solution that describes features of moving fluid such as velocities and temperatures (Chen and Srebric 2002). The CFD code directs computers to perform calculations. In order to confirm the credibility of CFD simulation, Chen and Srebric (2002) designed a procedure to verify and validate CFD code. In the verification phase, it identified the relevant physical phenomena like basic flow heat transfer to assess if CFD is capable to account for those phenomena; the validate phase demonstrated the linking ability of users and CFD code to conduct a simulation accurately; the report phase summarize the CFD analysis and assess its value and quality (Chen and Srebric 2002).

In the computation of turbulent flows models, the goal is to create a model to obtain quantities of parameter such as fluid patterns. However, the complexity of turbulence and the range of length usually exceed the capability of processing equipment. The primary approach in such cases is to create numerical models to approximate unresolved phenomena (Comas 2014). Stamou and Katsiris (2006)'s study has verified three mainly used CFD model— direct numerical simulation (DNS), large eddy simulation (LES) and Reynolds averaged Navier-Stokes (RANS) for indoor airflow and heat transfer. In their study, the former two models were rejected to be used for modeling office room, because the grid resolution of the first two models requires prohibitive computer resources which can't be achieved. Therefore, they chose RANS model and its variants to predict airflow velocities and heat transfer (Stamou and Katsiris 2006). Gousseau et al. (2011) conducted a comparison study of RANS model and LES model in predicting convective and turbulent mass fluxes. This study aimed at pollutant dispersion of an isolated building and the result showed RANS model can be applied in broader situations rather than LES model (Gousseau et al. 2011).

Recently, the recognition of environmental protection and energy consumption has been attached to design and post-occupancy operation. The energy consumed in building operations has a significant impact on cost and environment, which closely bonds with indoor climate. Building Performance Simulation (BPS) is being increasingly used in the earlier design phase. They allow designers to gain some information about energy performance. CFD integrated with building energy performance is studied by more and more researchers. According to Zhai and Chen (2005)'s study, researchers outlined a coupled Energy Simulation (ES) and CFD programs. Based on the principles and strategies they developed before, they designed a program by incorporating CFD into an energy simulation program (Energy Plus). This program made fully use of the implemental information by CFD and ES to increase the accuracy of simulation. Experimental data from four experimental facilities were used to validate this program and the validation shown this simulation could produce more accurate results than the separated simulations (Zhai and Chen 2005).

3.3 CFD and Chilled Beam System

In addition to the entire building energy simulation, CFD also has been utilized for specific air conditioning and ventilation systems to study and visualize their actual performance. Chilled beam system is one of them which has been equipped in many buildings. In order to determine how chilled beam systems affect conditioned room and test its performance, there are many studies using CFD to simulate an assumed scenario and compared with an actual environment.

The purpose of VEMPATI (2011)'s study was to investigate temperature distribution and air velocity pattern in a designed room fitted with two active chilled beams. The room geometry model was created by using GAMBIT 2.4.6 which is a general preprocessor for CFD. Key input data like the chilled beam type and the amount of primary air were acquired from the equipment suppliers. FLUENT software provided the CFD solution for this study. The results obtained from CFD simulations were validated by the velocity measurements. The uniform temperature distribution pattern was confirmed by comparing chilled beam with a traditional multi-cone diffuser (VEMPATI 2011).

In another study, an actual hospital ward case was analyzed to make sure the chilled beam system is better in controlling air path and preventing cross-infection. The simulation procedure is similar to VEMPATI (2011)'s study. The 'CFD-ACE' software package provided a platform from geometry model constructing, meshing and simulating to result viewing. But it was differentiated by building an actual temporary ward to conduct the cooling and heating tests. The results confirmed the advantage of chilled beam system in controlling indoor air path and verified the reliability of CFD simulation although there are still some inaccurate aspects (Devlin 2011).

The studies above verified the advantages of chilled beam system by simulating chilled beams' performance though CFD analysis. However, CFD is not a simple tool to use. Different parameter settings in boundary conditions lead to various results. Moreover, the airflow patterns under a chilled beam are not like a traditional diffuser. Therefore, it is important to find out the specific boundary condition settings of the chilled beam. This is what Mustakallio et al. (2005) did in his study. The special attention in this study was how different boundary condition settings affect the final result. The result showed a generic model without detailed knowledge caused an unrealistic air velocity and distribution pattern. Thus, some specific product boundary condition tools are needed in the future for more accurate CFD simulation (Mustakallio et al. 2005).

In these study cases, CFD was proven to be a useful and accurate tool to study airflow patterns and indoor environment. And it always starts with creating geometry model of the target building or room, which also leads to an interesting idea—what if the geometry models already existed?

3.4 CFD and BIM

It is hard to discuss geometry model and visualization without thinking of a popular and powerful tool used in the AEC industry—BIM. The 3D model created in BIM provides an intuitional view for designers and engineers to visualize the appearance, structure, orientation and other features of a building before it is actually built. Different applications have made it possible to expand BIM to cost estimating, construction sequencing, collision detection, facility management, etc. The data embedded in the model facilitates practitioners and consultants to conduct the performance simulation and energy monitor. Integrating BIM and CFD simulation has occurred frequently in recent studies to monitor and analyze indoor climate and energy consumption

O'Grady and Keane (2006)'s study found the biggest barrier in integrating CFD with building performance is the complicate CFD principles. Lacking of user-friendly software interface hindered the CFD simulation in conceptual design. Aiming at this problem, they developed an intelligent interface to combine BIM and CFD. All the geometric information in building information model was imported to this interface through Industry Foundation Class (IFC). The CFD solving algorithms was programed in this interface too (O'Grady and Keane 2006). This study provided an idea of combining BIM and CFD to conduct performance simulation. Based on Ham and Golparvar-Fard (2012)' s study, based on the deficiency of currently used energy performance simulation tools and sensing technologies, they created an image-based model incorporating CFD to simulate the indoor thermal environment. Then the deviations were identified between the actual and the simulation thermal environment.

They hoped this model could provide a new approach to identify areas where need to be retrofitted (Ham and Golparvar-Fard 2012).

Apart from these models and framework designed to simulate and visualize indoor thermal climate and energy consumption, Yoon et al. (2014) devised an automated CFD system with the help of BIM technology to apply to a green building rating system. This study mainly aimed at the ventilation simulation analysis, because the Certification Standard for New Building (BCA) attaches great importance to building ventilation strategy. Four major processes were introduced, BIM to CFD, pre-process, post-process and documentation. Specific automation algorithms were presented for the design of this system. The result showed this system increased work efficiency and reduce human-related errors for building design to acquire BCA certificate(Yoon et al. 2014). The integration of BIM and CFD is also used to study wind environment between buildings. According to Lee and Song (2010)'s study, in order to evaluate the wind environment among high-rise buildings in the early-design stage, they wanted to have an immediate method to use CFD simulate wind environment. A BIM-based CFD tool was used for the building model in design stage. Data like geometry, topology and semantic from BIM software can be export to Design Builder (CFD software) through gbXML file(Lee and Song 2010).

CHAPTER IV

SIMULATION WORKFLOW

This chapter explains each specific step involved in how to turn an original building information model to a simplified and suitable geometry model and how to use CFD to simulate Francis Hall indoor environment. The workflow is summarized from of trials and errors. The outline of the process is listed as follows and the flow chart is indicated as Fig.1.

1) Pre-processing
   - Model acquisition and correction
   - Model idealization
   - Test units
2) CFD Solving
   - Parameters input (materials and boundary conditions)
   - Mesh generation
   - CFD solving
3) Result Representation
   - Single office simulation
   - Third floor simulation
   - Entire building simulation

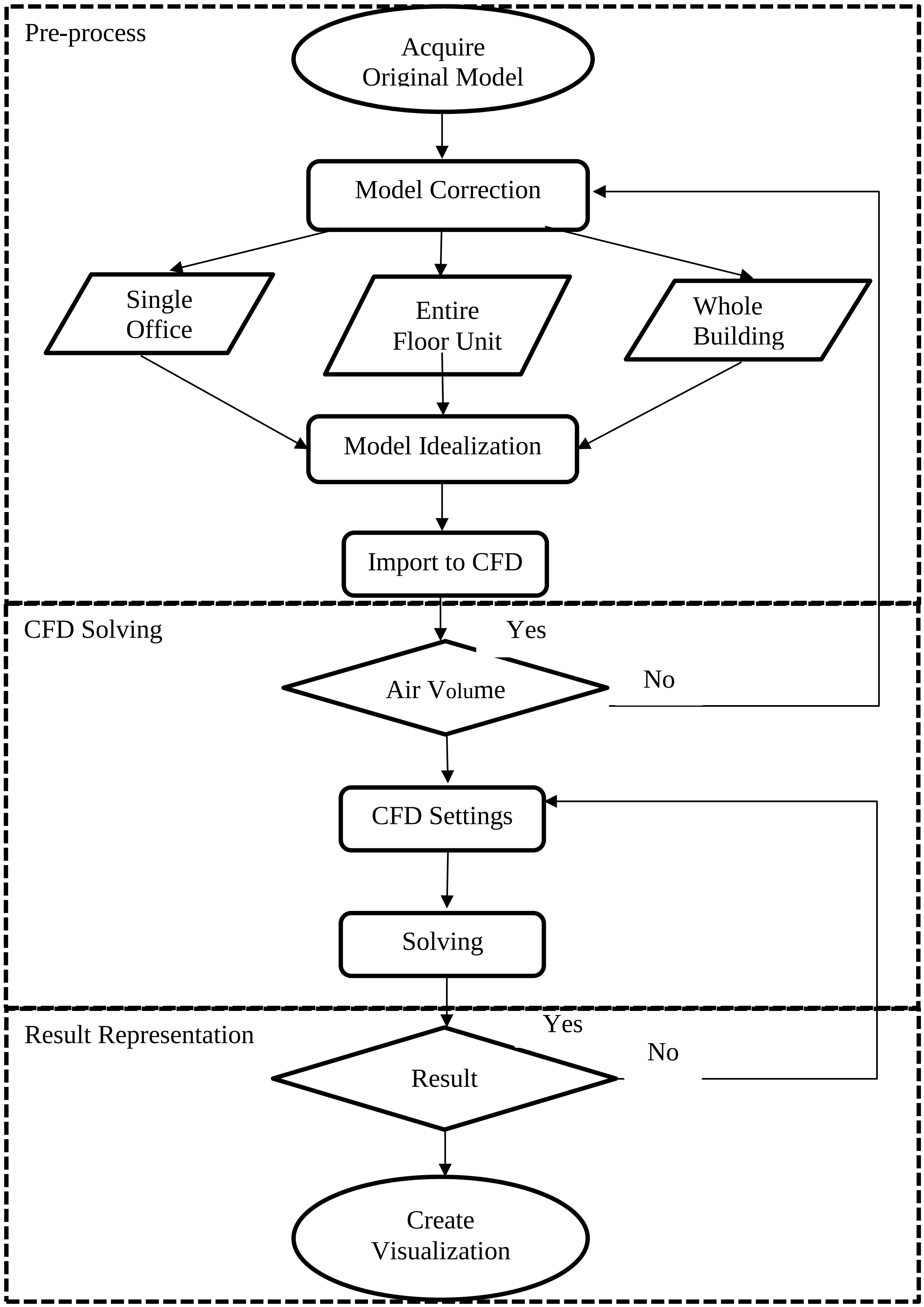

Figure 1 Simulation workflow

4.1 Pre-processing

The purpose of this stage is to get the original building information model ready for later CFD simulation. Actually, in most construction projects, the major function of BIM is visual representation. This technique makes the geometry model realistic and full of detailed instances, which also makes it complex and redundant for CFD simulation. Removing, simplifying and reorganizing are the key steps in this stage. After the preparation job has been done, the geometry models are going to be exported to Autodesk Simulation CFD.

4.1.1 Model acquisition and correction

The original model of Francis Hall was provided by the general contractor of building renovation project. They used Autodesk Revit as the BIM software. The façade configuration of this model is perfect. The type and location of other building components like windows, doors, stairs and ramps are almost the same as the actual building. Structural elements like beams, columns and the foundation are also revealed in the model. However, the inside layout of building is a completely mess because it is a model for renovation project. Thus, the inner layout is unable to be used directly for further steps, because the layout of the walls has an essential impact of indoor airflow traveling paths. This study requires the interior space to follow the realistic case. The first move was to restore the model to the actual situation of Francis Hall based on the shop drawings.

4.1.2 Model idealization

Model idealization means capturing design intent with the minimum amount of complexity (Autodesk 2015). Although a fully detailed model is necessary for directing construction and other purposes, only some of those objects are useful for CFD analysis. An idealized geometry model is important to optimize the simulation performance through reducing complicated mesh scenarios and saving computation time. The degree of acceptable complexity is the key consideration. For this Francis Hall case, removing, simplifying and adding instances in the model were the way to achieve model idealization.

Remove structural elements. The original building model has a complete structure design including slabs, beams, columns and the foundation which has to be removed. In a simple CFD simulation, the essential model objects are those elements which impact airflow travel path (walls, doors), generate heat (light, computers), transfer heat (windows) and condition and ventilate air (chilled beams). Most of the structure elements are hidden or covered by other architecture objects. For example, most columns are connected by walls and slabs that can be replaced by floors. It actually makes little difference to remove or substitute these structural elements in a CFD analysis. Fig.2 shows the comparison between original model and simplified model. The inner view is much cleaner in the simplified model. The auditorium is removed because its HVAC system is different and this study is not going to cover that part.

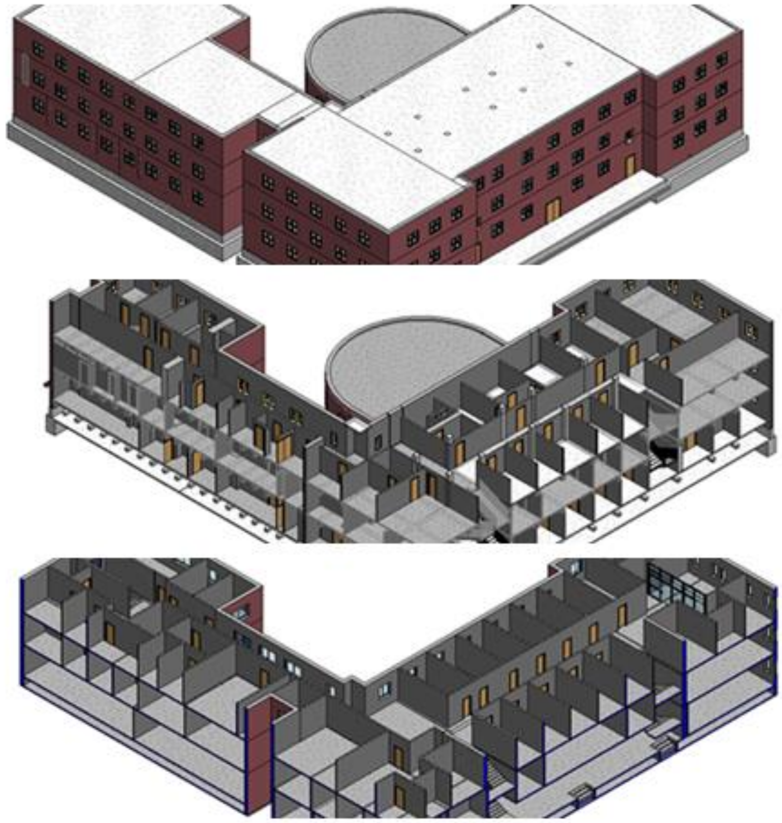

Figure 2 Original model and simplified model

Simplify components. Simplifying architectural components is also necessary to idealize the model. Usually, the components instances provided by Revit are full of details. This is good for visual representation and construction, but not for CFD simulation because of the meshing. Meshing is a process that divides every piece of module into small cells (called grid). Then, some important parameters (pressure, velocity and etc.) are calculated in each cell. An over-detailed model object adds extra grids and consumes more time when solving. Another issue for detailed model objects is the selection problem in CFD. This will be explained later. Therefore, it is better to use simple shape configurations to replace detailed model objects. Fig.3 shows the comparison between original and simplified window.

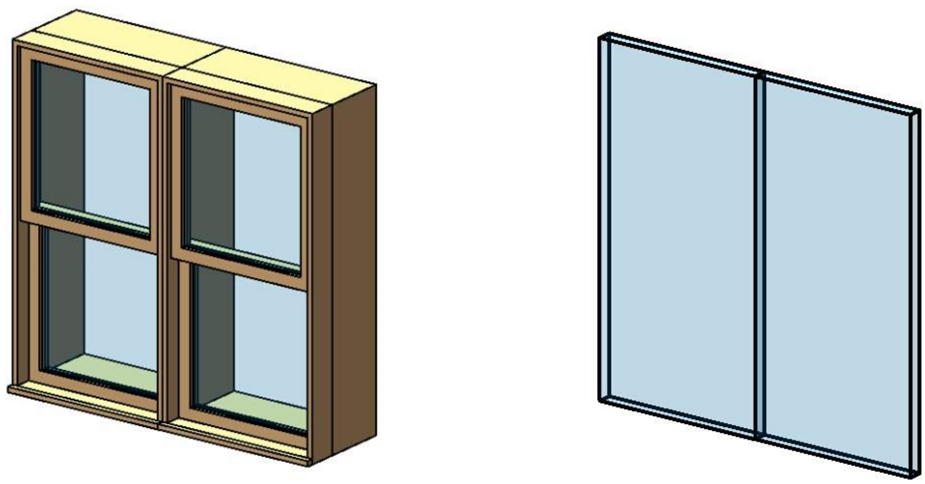

Figure 3 Window model comparison

Add MEP components. Chilled beams and exhaust fans are the essential mechanical components in Francis Hall that help to keep good indoor air quality. However, the provided building model doesn't have any HVAC components. In this study, the chilled beam family was created by the author based on the configuration of the product user's manual. Fig.4 shows the actual chilled beam from the user manual and the chilled beam model in Revit. Simple volumes and surfaces were used to represent the geometry and their performance will be defined in CFD simulation. Exhaust fans family was downloaded from Revit City. The layout of each chilled beam and exhaust fan are assigned according to the HVAC shop drawings.

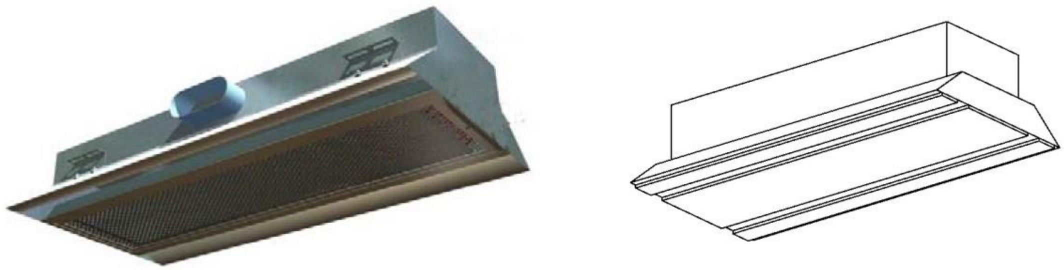

Figure 4 Actual chilled beam and chilled beam model

4.1.3 Test unit

In order to achieve a comprehensive understanding of the feasibility and practicability of combing BIM and CFD, this study simulated three models of different building sizes and levels of detail while targeting at specific preference. Fig.5, Fig.6 and Fig.7 show these model units.

Single office model provides the most detailed information and has everything almost the same as a realistic office, especially for those equipment that generate heat such as lighting and computers. The office appliance layout and the chilled beam working pattern are also shown in the simulation. The purposes of the model are to visualize the airflow traveling patterns and temperature variations and how they are affected by heat sources.

Entire third floor model deals with a general situation. Choosing the third floor is because there are two exhaust fans on the top of the roof that work as the air return terminals for the entire building. This test unit simulates how the air travels from the corridor chilled beams to the exhaust fans, as well as the temperature distribution in the third floor. Indoor layout and chilled beam location are the major concerns in the model.

Other features such as lighting and human activities are not considered in this unit even though their heat generation may lead to a slight difference in temperature.

Entire building model is used to simulate the entire building airflow and temperature distribution situation. Some complex factors such as elevator, mechanic room and other structure instances are reduced, but the essential components are kept same as the third floor unit. All the necessary model objects were placed according to the building renovation shop drawings in pursuit of authenticity. This model aims at testing if the CFD analysis can solve the large building simulation and how long it will cost in a regular personal computer. This is important to know if people want to use CFD simulation for other real building cases.

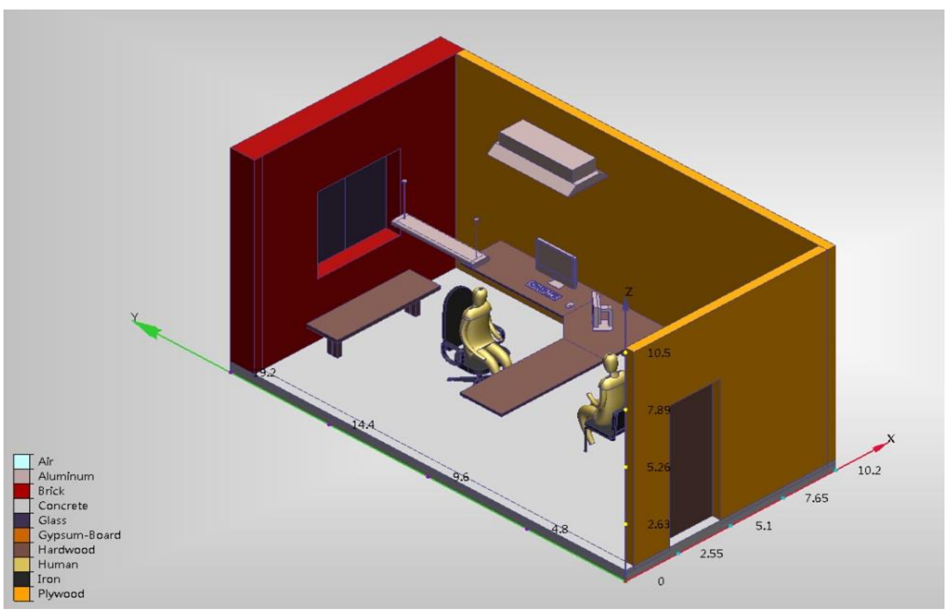

Figure 5 Single office unit

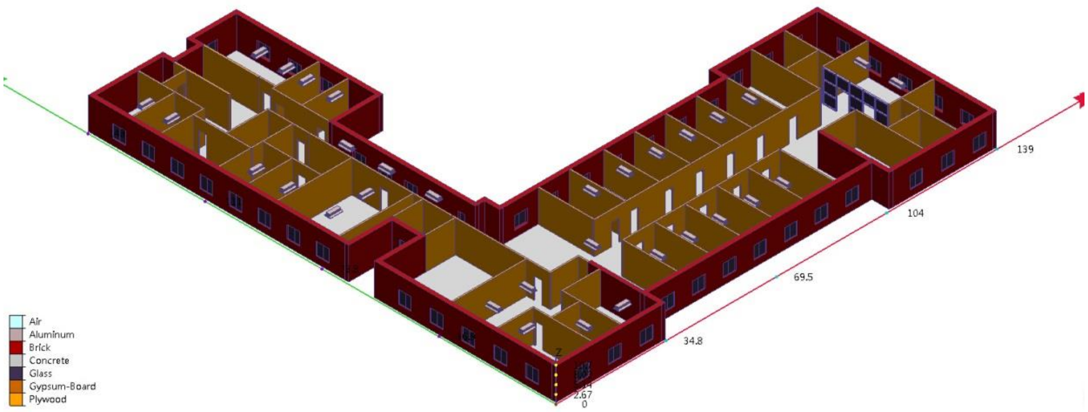

Figure 6 Third floor model

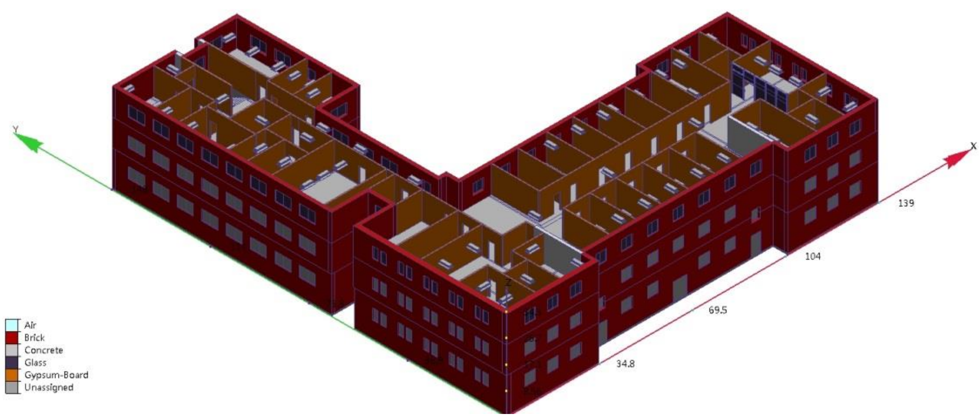

Figure 7 Entire building model

4.2 CFD Solving

This section is going to introduce all the preparation and setting works in Autodesk Simulation CFD. The reasons why to use this software is because: 1) both BIM and CFD software are from the same company and the interoperability is positive; 2) this software has a user-friendly interface that is suitable for those who are not experts in CFD. It is very straight-forward to use this software for some basic analysis without involving some complicated CFD principles. In this section, the major attention is to explain different settings in each step and where to get information for those inputs.

4.2.1 Parameters input (materials and boundary condition)

The only information of the exported geometry model in CFD is just the location and the connection between each model object. Other information and data need to be defined by users. Assigning material is the first step. It defines simulation domain components and impacts the simulation physics and its results (Autodesk 2015). Fluid and solid were the only two types of materials in this study. Everything in the model except indoor air was defined as solid materials like concrete, wood and etc.

After all the model objects have been defined, the next step is to define boundary conditions. Boundary conditions are prescribed values that specify fluxes and define how they interact with geometry model and its environment. It includes: 1) identifying the locations or surfaces of boundaries; 2) assigning information at the boundaries. As the test models, this study relies on the basic settings of boundary conditions in order to avoid a longer solving time and uncertain mistakes. But the core content which is the air ventilation and conditioning distribution pattern will be simulated.

Materials in this model were defined based on the information from the Revit model. Aluminum was used for some equipment instance generally like computer and lighting. For the data of boundary conditions, some of them came from MEP shop drawings (airflow volume, temperature), the rest (film coefficient, heat generation) were based on online information. Fig.8 reveals the model in material and boundary conditions selection interface. The detailed information of materials and boundary conditions are show in table 1.

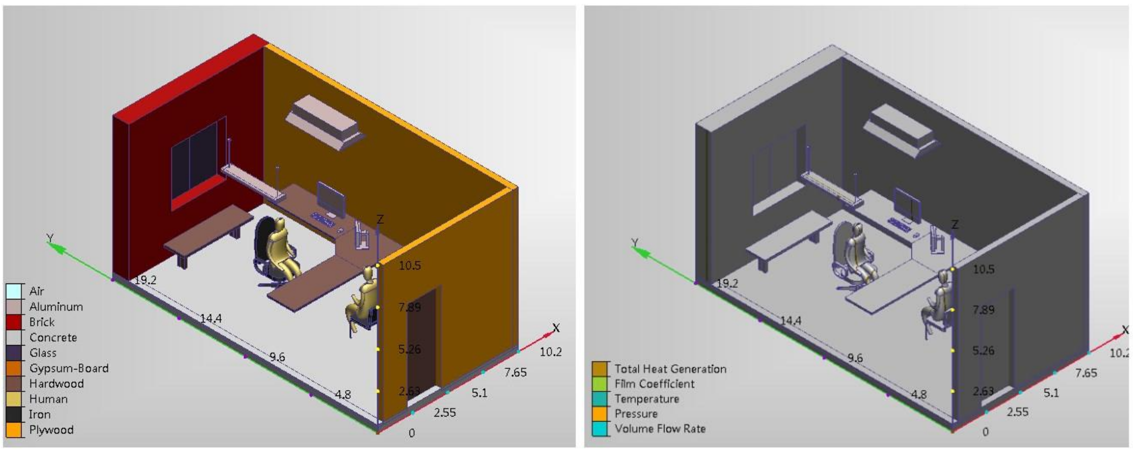

Figure 8 Material and boundary conditions selection

Table 1 Materials and boundary conditions

Name	Material	Boundary Condition	Function	Unit	Value
Air volume	Air, variable	---	---	---	---
Exterior walls	Brick	Film Coefficient	Heat Convection	BTU/ft2/h/R	0.36
Interior walls	Gypsum-Board	---	---	---	---
Window	Glass	Film Coefficient	Heat Convection	BTU/ft2/h/R	0.27
Door	Hardwood	---	---	---	---
Roof	Concrete	Film Coefficient	Heat Convection	BTU/ft2/h/R	0.3
Floor	Concrete	---	---	---	---
Chilled Beam	Aluminum	Volume Flow Rate & Temperature	Air inlet Temperature	CFM Fahrenheit	100 60

Table 1 Continued

Name	Material	Boundary Condition	Function	Unit	Value
Exhaust Fan	Aluminum	Pressure	Air Outlet	Psi	0
Computer	Aluminum	Total Heat Generation	Generate Heat	W	200
Lighting	Aluminum	Total Heat Generation	Generate Heat	W	100
People	Human body	Total Heat Generation	Generate Heat	W	60

4.2.2 Mesh generation

Meshing is the next step that means the entire model is divided into discrete cells and these cells are called mesh. Then, various parameters (like temperature, velocity, etc.) will be calculated in these cells iteratively together. The quality of mesh is one of the critical factors that impact the simulation speed and accuracy. The finer the mesh (more cells), the better the scope of calculation, and the more accurate the result is, because it provides detailed scope to study parameters. However, there is a dilemma that as the number of cells increase, it takes a longer time to solve the simulation. The key point is to find an appropriate mesh that coordinates the solving time and results' accuracy. In Autodesk Simulation CFD, the Automatic Mesh Sizing function makes meshing very simple. The defined mesh accurately shows the details of the geometry model. Another function called Mesh Adaptation further optimizes the mesh definition. The basis of this function is using the results in the previous simulation cycle to refine the mesh. This study took the default automatic meshing to test the basic general situation. Fig.9 shows the mesh in single office case.

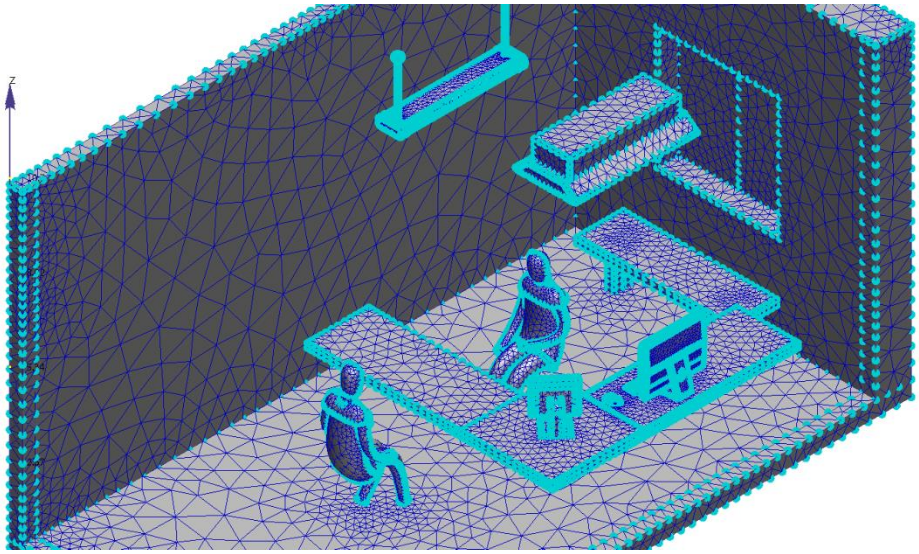

Figure 9 Meshing in single office

4.2.3 Solving

After all the settings have been defined correctly, the next stage is CFD solving. The CFD solving principles are not going to be involved in this section because it is not the emphasis of this study. This part will just introduce some regular setting for AEC industry study and how they help to solve the simulation. This Francis Hall case is based on the following key assumption:

- Solution model is Steady-State
- 100 iterations to run
- Incompressible flow
- Enable heat transfer and radiation

There are two different solving models in Simulation CFD. Steady-state model assumes all the parameters have been running for a long time without any changes or interruptions; Transient model solves simulations that some inputs are changed for a specific amount of time. Once a steady-state analysis is started, all the variables are solving iteratively to reach a stable point where the flow is at a state of equilibrium. Meanwhile, the number of iterations shows how many times the calculation algorithm has been run. Besides, the heating & cooling equipment in the building cause the heat transfer through convection, some heat generation sources also have heat transfer with surrounding air through radiation.

This study assumed all the HVAC equipment has been running for a long time. 100 iterations was the default setting that can solve most scenarios. And the season for this study was summer. All the assumptions reflected a realistic condition of Francis Hall.

4.3 Result Visualization

The simulation results of three test units will be shown and interpreted in this section. Simulation CFD provides numerous ways to visualize different variables. They are helpful to understand of how chilled beam system work and how they affect indoor environment in Francis Hall. Airflow distribution and air temperature variation are the major concerns in this section. Three important functions are going to be used to visualize the results.

Result panels is the primary tool for visualizing the results by using two-dimensional cutting planes. According to the coordinate system on the geometry model, users can choose planes of any positions to see temperature or velocity variations.

Vectors are usually added on those planes to clearly indicate the airflow directions. This function helps to study the temperature distribution on a specific panel.

Particle traces is a powerful tool to visualize flow movement. Airflow is shown as a pattern of lines or ribbons that moves from the air supply vents to the rest spce of building by following the fluid dynamic principle. It is like an injected dye stream in the flow. Another important function of particle traces is to make the animation to show the fascinating dynamic effect of airflow distribution.

Iso surfaces is the three-dimensional visualization tool that shows the physical shape and values of the flow characteristics. It is useful to see the airflow and temperature distribution in the entire indoor space. The Iso-surfaces shape indicates the specific velocity magnitude and the colors mean the static pressure at the corresponding locations.

4.3.1 Single office result

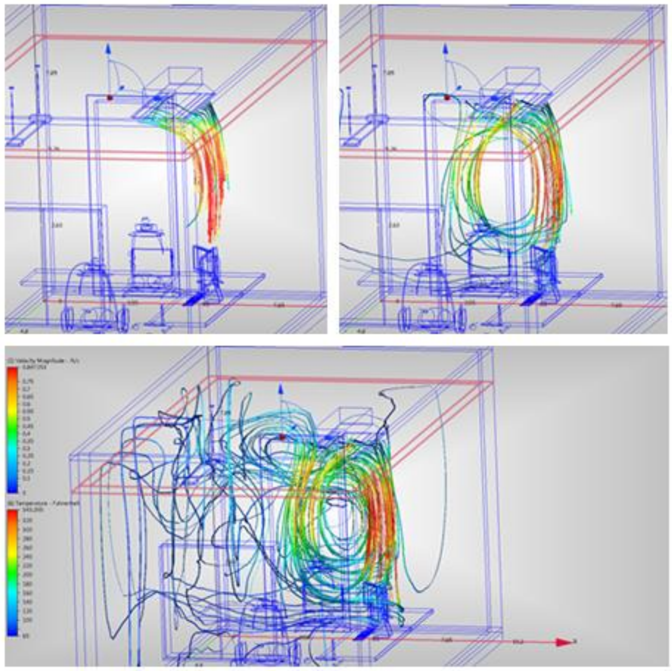

Figure 10 Air traces in single office

Fig.10 shows the air trace in a single faculty office when the chilled beam is working. The different colors in the ribbons indicate the temperature of air. The cooling air supplied from chilled beam become a little warmer when traveling around heat sources like computers and human. Then some of the air will return to chilled beam and go through the cooling coil and form a circle in the pictures; other air will condition the rest space in the entire room. Based on some general information about the chilled beam, this simulation makes sense to reveal the airflow pattern of chilled beams in a coarse level.

4.3.2 Third floor result

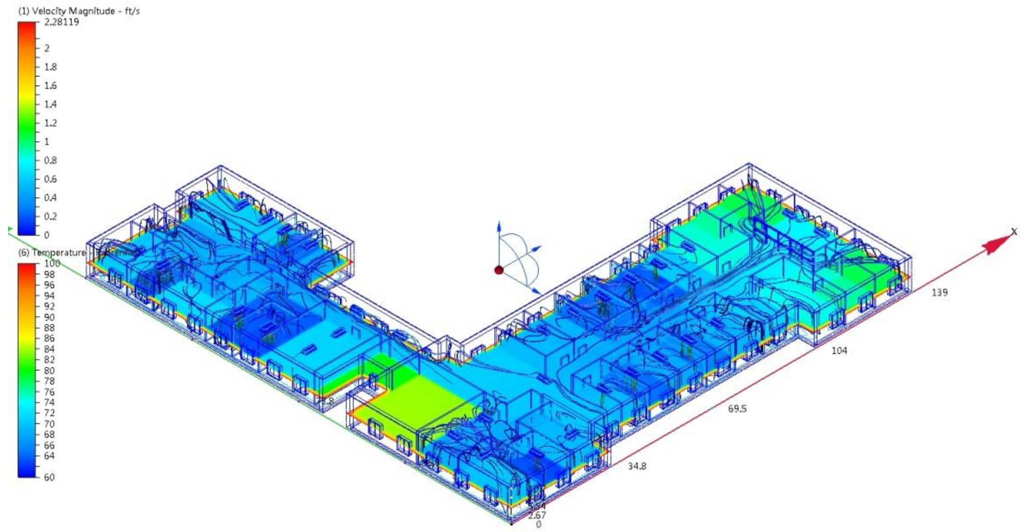

Figure 11 Result panel of third floor model

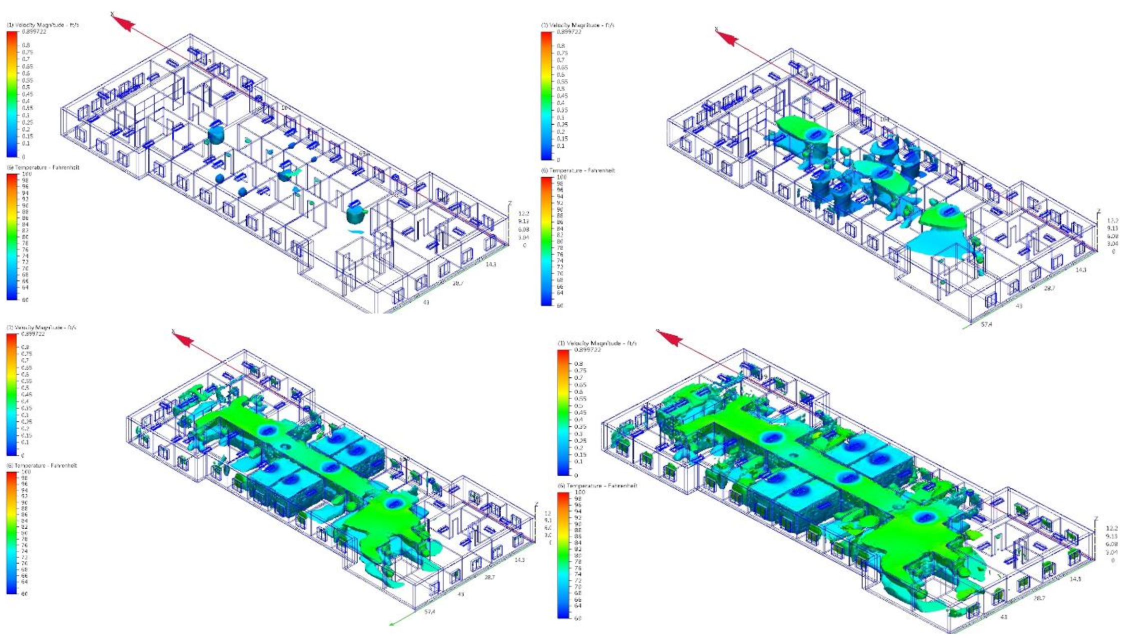

Figure 12 Iso-surface result of third floor

Fig.11 and Fig.12 present two different ways to see the simulation. In the panel view, the temperature is showed in a panel. The rooms equipped with chilled beams have a lower temperature that the rooms without chilled beams. Furthermore, the color on the exterior walls is red. This means the outdoor air is 100 Fahrenheit that simulates the outdoor temperature. Fig.12 presents the iso-surface view of the result. These images imitate how the air comes out from the chilled beams and gradually occupies the entire indoor space. In this simulation, three chilled beams in the hall way and four chilled beams in the faculty offices were activated. That is the reason why all the cooling air is supplied by these seven chilled beams. In the last image, there is a little amount of air that comes out of the building through two exhaust fans. This basically simulates the actual scenario of air circulation in third floor

4.3.3 Entire building result

The entire building model simulates all the chilled beams in the building and also adds large heat source such as BIM cave and BIM lab. Unfortunately, this study didn't get the final simulation results because of the calculation capability of a personal computer. After running this simulation for more than 24 hours, the laptop was crushed. This study couldn't show any simulation results of the whole building test unit.

CHAPTER V

PROCESS TIMING AND EVALUTION

Last chapter introduced the workflow of applying the building information model as the initial geometry model and simulate the indoor environment of Francis Hall. Much time has been consumed in optimizing the model and making it efficient for CFD simulation.

However, this working process is provided just based on author's personal experience, which lacks of supports from data and other information. In order to prove the necessity of every step and make this research convincing, the author went through the entire pre-process according to the workflow described above. Then, the time consumption of each step was recorded as important quantitative measurements. After that, the time consumed in CFD solving process between the original model and the simplified model was also recorded.

The basic conditions of recording job are described below:

- All the entire re-do work is finished by the author.
- All the time recordings are under a normal environment without any disturbance.
- A different computer is used to work on this process and there are no related files and documents in this computer.
- All the settings of original model and simplified model remain the same

5.1 Pre-process Time Consumption

Considering the different emphasis of three model units, this timing process mainly focused on the single office model and the entire floor model. The whole building model test unit was omitted here because the computer's ability is incapable to simulate this massive volume model. The actual timing sequence of these two models was a little inconsistent with the standardized workflow explained in last chapter. This is because the pre-process works for two models were independent of each other. They were isolated from the whole building model first and then re-organized, which was different. But the other steps completely followed the workflow. Table 2 and Table 3 list all specific steps and their time cost.

Table 2 Single office pre-processing time

Model Preparation	Explanation	Time Consuming
Isolate CFD test unit	Delete and create office	10 min
Re-organize interior layout	Add furniture, light and other components	12 min
Simplify objects	Simplify furniture, lighting and other components	19min
Add HVAC	Create family	29 min
Export and check	Air volume lost	13 min
Total time	-	83 min

As shown in the table, most of time during this process was spent on simplifying component model objects and creating the chilled beam model. For the single office model unit, heat sources and human activities are the indispensable elements, because the airflow and temperature are easily affected by them. The specific method to simplify these components objects was to hide some less important structure or replace some complicated structures with simple geometry. In this model, many unnecessary parts were removed from the models of window, chair, computer, door and light. Some of them were replaced by simple volumes. After that, creating the chilled beam family was another step that consumed much time in this process.

Table 3 Entire floor pre-processing time

Model Preparation	Explanation	Time Consuming
Isolate CFD test unit	-	6 min
Re-organize interior layout	Floor & Ceiling	7 min
	Interior walls	54 min
	Doors	6 min
Simplify model objects	Windows	11 min
Add HVAC	Create family	29 min
	Place chilled beams	12 min
Transfer and Check Air volume	Find and correct problem	29 min
Total time	-	154 min

Comparing to the single office model, most of the time has been spent on re-organizing interior layout and checking air volume in the entire floor model. This is because the primary model interior layout was totally different from what the building has now. All the interior walls and doors were placed in this stage. When importing the model to CFD Simulation, the air volume was lost in the CFD Simulation at first. The reason was the windows in the original model could not be recognized correctly in CFD Simulation. Thus, it took about half hour to fix this problem.

5.2 Simulation Time Consumption

In this part, the time consumption between the original model and the simplified model was compared in each step, from material selection to simulation solving. In addition, another important data was collected and compared in this stage—number of model objects in CFD Simulation. As mentioned before, every piece of model object in CFD is selectable. This means some components in Revit can be selected as one model object, but their ingredients are all selectable in CFD. The number of these objects can be used as an indicator to show model's complexity. Time consumption and used model objects in each step are shown in table 4 and table 5.

Table 4 Single office comparison

Model unit	Original model		Simplified model	
Measurement	Time	Number of objects	Time	Number of objects
Assign materials	11 min	262 objects	7 min	66 objects
Assign boundary conditions	4 min	16 objects 4 surfaces	3 min	14 objects 5 surfaces
Meshing	-	1,081,416 mesh cells	-	364,168 mesh cells
Solving	83 min		39 min	
Total time	98 min		49 min	

The biggest difference between original model and simplified model is the complexity of components model. This can be directly revealed from the number of objects. Because the single office is a small model with limited space, the time difference spent on first two steps was not obvious. However, those delicate model objects in Revit massively increased the available model volumes in CFD Simulation. Meshing those unnecessary volumes resulted in large numbers of extra cells. The cell number of simplified model was about one third of the original model. Therefore, the time spent on solving original model was about twice as much as the simplified model.

Table 5 Entire third floor comparison

Model unit	Original model		Simplified model	
Measurement	Time	Objects number	Time	Objects number
Assign materials	18 min	831 objects	10 min	272 objects
Assign boundary conditions	17 min	300 surfaces	7 min	184 surface
Meshing	-	3,610,329 mesh cells	-	2,076,207 mesh cells
Solving	121 min		74min	
Total time	156 min		91 min	

When solving the entire floor unit, the time gap between two models was obvious. The original model used for comparison was isolated from the whole building model.

After fixing the model airtight problem and adding some chilled beams, the model was directly imported to CFD Simulation. The simplified model was processed by following the workflow. As shown in the table, the number of objects and boundary condition objects varied widely. This made the time consumption on selecting those objects different. One thing need to be noticed is the mesh cells. The amount difference is not large as single office model. This is because the number of mesh cells is heavily affected by delicate small model objects and this model unit didn't have many small model objects like single office.

5.3 Result Analysis

Based on the timing data, the first question of this study can be answered. On the software level, these two tools can work together. There is no problem to transfer the building information model to CFD software. But on the practical level, the building information model cannot be directly used if people want to study indoor environment. In this Francis Hall case, every test model unit has the same problem which is the air volume in CFD model is missing. That is the reason why time has to be spent on checking and fixing the model airtight problem. It took 13 minutes to fix this problem for single office model and 29 minutes to fix the entire floor model. Even in the model comparison section, the original models were still modified in order to become airtight and continue this study.

In order to prove the necessities of model pre-processing, after fixing the model airtight problem, this study compared the model which didn't go through the pre-process with the processed model in the aspects of time consumption and model complexity. According to Table 4 and Table 5, the simplified models showed a simpler geometric structure which created less model objects and mesh cells. Less model objects indicate less manual work on defining materials and boundary conditions; less mesh cells mean light workload in simulation and a shorter solving time. This also proved it is necessary to process the building information model not only because of the flaws and defects, but also for a quicker and easier way to solve the problem.

CHAPTER VI

DISCUSSION AND SUGGESTION

Last two chapters introduced the basic workflow of using building information model for CFD simulation and the time consumption of each step. They confirmed the integration process is not simple and straightforward. Barriers and problems make this work over-complicate and time-consuming. This chapter is going to explain what are those problems and barriers during the collaboration. Are these problems caused by software itself or data collection and how to solve them? Then, this study will give some suggestions for future development and improvement.

6.1 Barriers in Integration

The original building information model is not suitable for CFD simulation usage. A lot of editing and preparing work needs to be done at the beginning of this study in order to make it ready for the next stage. Problems in this collaboration stage are going to be discussed in this section.

6.1.1 Model defects

Inaccurate model was the first problem in this study. The original model provided was a renovation project. It was not built based on the current indoor layout of Francis Hall. the type and location of some interior walls were not corresponding with the real case. In most real projects, buildings are not constructed by completely following the initial design and drawings. Many changes happened during the construction phase. However, most building information models are created based on the original drawings.

There must be some differences between the actual building and the 3-D geometry model. Problems may exist due to the differences because the CFD geometry model requires a precise interior or exterior instance layout to show the airflow. The only way to solve this is to modify the model manually.

Redundant element means some model objects in building information model that are unnecessary for CFD simulation. Architectural, structural and mechanical elements make up a complete building information model. And all the major information of the building is embedded in those model objects.

However, in the CFD simulation process, much unnecessary geometry data only means more burden for computational analysis. For example, sometimes the MEP system elements like ducts, pipes and vents largely expand the data size of the model which makes it incredibly complicated. The major focus of CFD simulation is the fluid flow in the space between solid objects. What really matters are air terminal equipment models where fluid comes out and goes in and interior wall objects that obstruct and change the direction of airflow. Some structural and mechanical objects that barely affect airflow traveling path should be eliminated from the geometry model. In this Francis Hall case, the beams, columns and the foundation don’t play any roles in CFD simulation, which should be removed.

HVAC element missing is another important issue in the Francis Hall model. Basically, most building information models for visual representation purpose usually do not have mechanic elements. But some of the HVAC equipment is an essential component in CFD simulation to define where the air comes from. In this case, chilled beams were added based on the design shop drawings. Furthermore, as the chilled beam system is not widely used in many buildings, a new family was created in Revit to indicate the chilled beams. In order to reduce the simulation workload and simplify the model, this family was created by using simple volumes and surface to represent their existence geometrically.

6.1.2 Over-detailed model instance

In Revit, every object is made up as close as possible to the actual instance in order to achieve a better visual effect. This is also helpful to reveal the information for different items. However, CFD analysis does not need the extra information for the simulation. Over-detailed model objects will increase the time to define parameters and the time to solve simulation.

Material define is the first step for CFD analysis. Every volume or surface created in Revit is selectable in Simulation CFD. These items are used to define inputs and extract results. The number of them is a factor in simulation solving time. In order to simulate the indoor environment as realistic as possible, heat generation sources like human bodies, lights and electronic equipment are supposed to be added in the model. The better appearance they have, the more intricate structure the model has. Even this makes it looks realistic in the model, it also means heavy workload for analyzing and calculating the airflow.

During the pre-process stage in CFD application, every item in the model has to be assigned with materials. This is the first steps and also the prerequisite for running the final simulation. Because every piece of elaborate component items is selectable, these tiny useless pieces are “debris geometry” and each of them needs to be assigned material. If the building information model fills with these items, the workload of people and machine is incredibly large. However, “simple delete and replace” way to simplify those items is not applicable, as most of them are more complex than some simple geometric shape. Fig.13 and Fig.14 indicate this problem in different models.

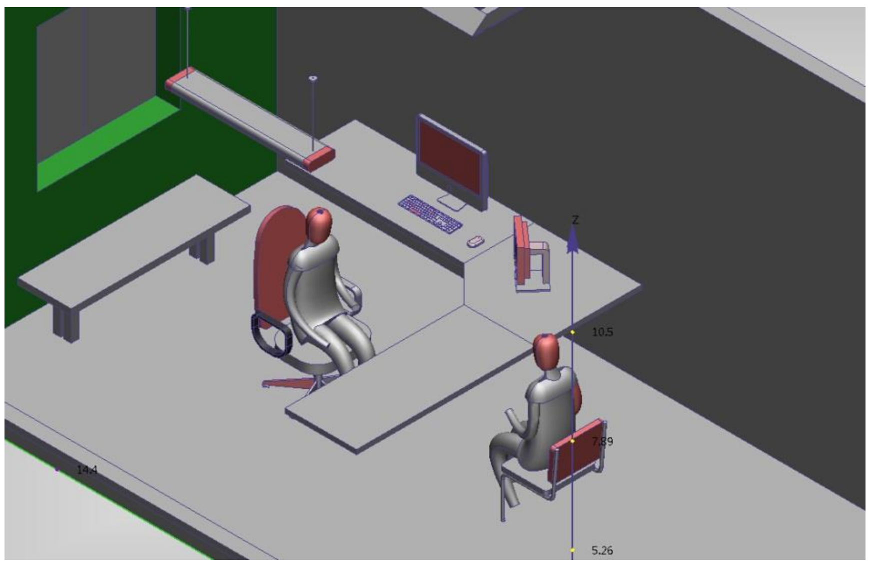

Figure 13 Selectable objects in the single office model

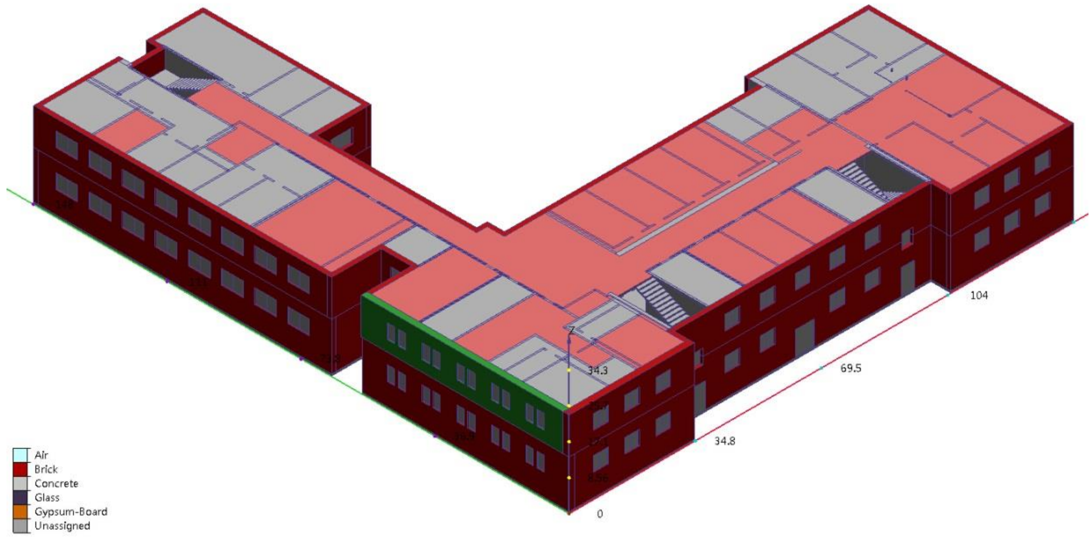

Figure 14 Selectable objects in the entire building model

Mesh generation is another incentive to simplify model objects. The function and principal has been introduced before. Using simple model characters to substitute delicate model objects is an effective way to facilitate simulation. In this case, windows and doors are indispensable elements because they have different heat transfer rate that need to be defined individually. But components like mullions, frames and glasses in the simple window object model are totally unnecessary for CFD analysis. This Revit family was edited to show only a piece of glass in a coarse detailed level. Other object which is hard to modify such as computers are replaced by cube volumes. Figure 15 shows the mesh conditions in windows of different detailed level.

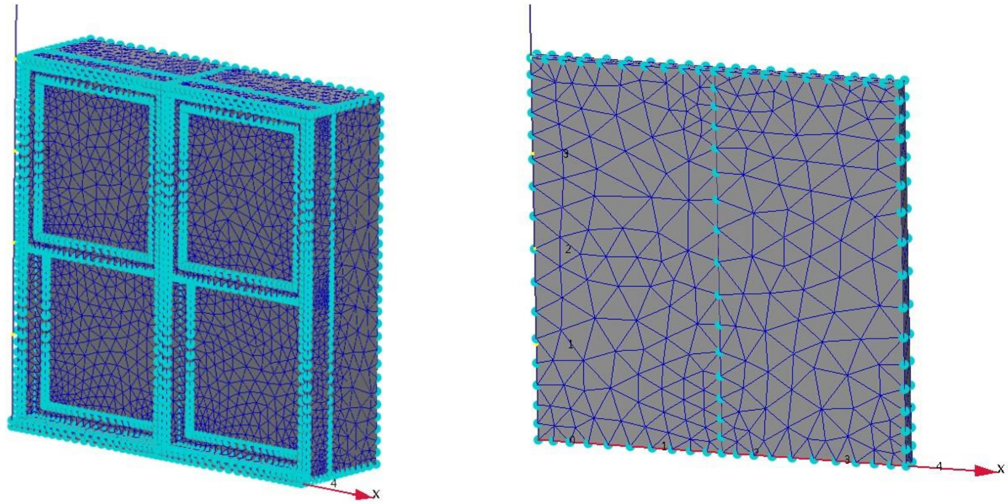

Figure 15 Mesh on windows

6.1.3 Model airtight problem

So far, this is the biggest and most time-consuming problem that affects the interoperability between two tools in this study. Defining material for every object in the geometry model is the first step to start the simulation. Solid model objects like walls, doors and floors are created in Revit. Each single piece becomes selectable and can be chosen directly after they are imported in Simulation CFD. As for the fluid material like air, they are created automatically in Simulation CFD as long as the air volume is fully enclosed inside Revit. But if there are some tiny gaps in the exterior surface of the model, it is very likely the air volume will not be generated. The air inside of the building is the core material studied in this research.

Although most of the models are precisely based on design drawing, some problems still exist. For example, two different walls are not connected on a strict straight line, tiny gaps occur between floors and ceilings. It is not difficult to check the envelop for a small size model, but when dealing with the entire building model with a large numbers of windows and doors, problems cannot be easily solved, because these problems are difficult to find out

In this Francis Hall case, this problem occurred every time when transferring Revit model to Simulation CFD. The author believed the unsealed building information model is not the major cause but the software itself, especially when transferring some large intricate model. Although tutorials provide solutions like using Geometry Tool or generating mesh to detect where the gap is, this did not help to solve the problem. In the end, after changing almost all the original exterior model instances, this problem was solved. There is still no clear answer to this problem. Fig.16 shows the air volume in third floor model.

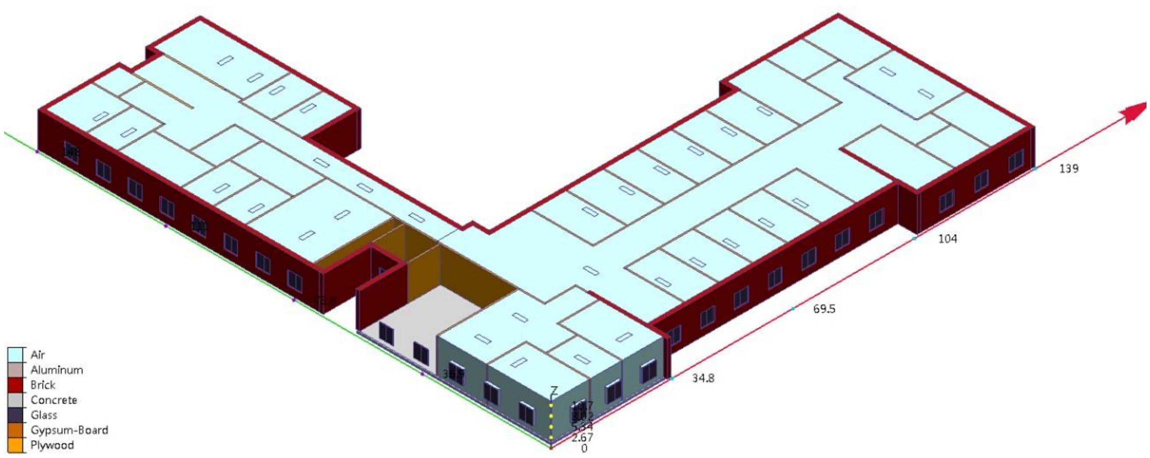

Figure 16 Air volume in the model

6.2 Barriers in Simulation

Computational fluid dynamics is a complicated subject that uses numerical analysis and principal from fluid mechanics. Although Autodesk Simulation CFD hides the intricate algorithm calculation process and turns those specialized settings to understandable and approachable interface for normal users without background, parameters input is still a problem when the actual ventilation and air conditioning situation is complicated. The setting problems largely affect the accuracy of the finally results of CFD simulation. This section will discuss these problems in CFD operating environment.

6.2.1 Special HVAC system

Usually, a standard indoor CFD model always comes with corresponding air inlet and outlet vents. It shows the airflow in a straightforward way. But a chilled beam supplies primary air mixed with secondary air going through cooling coil to the room and the secondary air comes from absorbing surrounding used air. This means one chilled beam have both air inlet and outlet. This is how airflow travels in a small section. For the entire building, indoor air is expelled through two exhaust fans. Thus, this study uses three model units in different scale to show a comprehensive airflow distribution situation.

The single office unit focuses on the airflow patterns in a small room; the emphasis of third floor unit is to figure out how airflow travels from chilled beams to exhaust fans in the corridor space; the entire building unit is used to visualize the overall airflow and temperature distribution. Different preferences make this testing process full of obstacles when setting parameters.

6.2.2 Applicable boundary conditions

Setting boundary condition is not an easy thing in simulating indoor environment, especially for construction consultants without a solid background in Fluid Dynamics. Actually, as Simulation CFD is a tool applied in many areas including aerospace, automobile, building, electronics and medical, there are many different types of boundary conditions dealing with different situations. Determining what are the meanings of different boundary conditions and which of them are related to construction industry needs a profound knowledge in mechanical engineering.

Much time of this study has been used to understand what the meaning of each boundary conditions and what scenarios they use for. Table 6 below shows some detailed information of each boundary condition. This can be a primary guide for understanding what their meaning and purpose. In this study, there are five boundary conditions are used to mimic the actual indoor environment. They are the most important factors that affect indoor airflow pattern and temperature variation.

Table 6 Boundary conditions and use

Boundary Condition	Objective	Category	Construction Use	Apply Scenario
Velocity	Surface	Flow	Yes	Inlet condition
Rotational Velocity	Surface	Flow	No	Rotating object surrounding by a fluid
Volume Flow Rate	Surface	Flow	Yes	Commonly used inlet condition
Mass Flow Rate	Surface	Flow	Yes	Commonly used inlet condition

Table 6 Continued

Boundary Condition	Objective	Category	Construction Use	Apply Scenario
Pressure	Surface	Flow	Yes	Mostly used as outlet condition
Temperature	Surface	Heat Transfer	Yes	Heat transfer analysis, usually with inlet condition
Slip/Symmetry	Surface	Flow	Yes	Building wind environment
Unknown	Surface	Flow	No	Mostly for outlets of compressible flow analysis
Scalar	Surface	Flow	No	Represent the concentration of the scalar
Humidity	Surface	Flow	Yes	Represent the relative humidity
Quality	Surface	Flow	No	Water vapor percentage of the steam
Heat Flux	Surface	Heat Transfer	Seldom	Heat value divided by area Exterior wall heat transfer
Total Heat Flux	Surface	Heat Transfer	Seldom	Total heat value Exterior wall heat transfer
Film Coefficient	Surface	Heat Transfer	Yes	Natural convention heat transfer
Radiation	Surface	Heat Transfer	Seldom	Radiative heat transfer
External Fan	Surface	Flow	Yes	Varied inlet flow rate condition
Current	Surface	Heat Transfer	No	Heat generation by electronic current through a metal
Voltage	Surface	Heat Transfer	No	Heat generation by electronic current through a metal
Total Heat Generation	Volumetric	Heat Transfer	Yes	Heat-dissipating components Heat load divided by volume

Table 6 Continued

Boundary Condition	Objective	Category	Construction Use	Apply Scenario
Transparent	Surface	Heat Transfer	Yes	Radiative heat transfer through transparent material
Periodic	Surface	Flow	No	Axial or centrifugal turbomachine
Heat Generation	Volumetric	Heat Transfer	Yes	Heat-dissipating components Heat load not divided by volume

6.2.3 Large-scale model simulation

The whole building indoor airflow simulation is a challenging work in this study because this building size has never been done with Simulation CFD before. Usually, the entire building model is used for simulating wind environment and energy analysis. These studies simplified building as much as possible. However, the indoor environment study for entire Francis Hall has kept most of the actual building configuration and interior layout. All the chilled beams and other major components are based on the actual design. This heterogeneous structure inside of building incredibly increased the computational work. The entire building simulation failed after running the program for more than 24 hours because the laptop was crushed

Although there are other CFD simulation strategies to reduce total calculation time such as coupled multi-zone-CFD model, this method could not work out due to the non-uniform inside layout and chilled beam location and the Simulation CFD is incapable of this function itself. Overall, it is not feasible and practical to simulate the large complex whole building indoor environment like Francis Hall.

6.3 Suggestions and Future Development

Currently, the collaboration of Autodesk Revit and Autodesk Simulation CFD is still a complicated and time-consuming job. The original building information models always have problems. They are either too complicated or short of important components. Although they can be used as the geometry model, the time spent on modifying and simplifying original model is a serious problem to consider. If there are some functions to optimize their interoperability, it will boost the interoperability to use them together. The following sections introduce some suggestions concluded from this study.

6.3.1 Model simplify function in Revit

Autodesk Revit provides a platform to create building geometry model with comprehensive information. It is a good tool to direct construction and help with visual effect. But when associated with other tools, part of information could be unnecessary and redundant. The detailed model objects sometimes greatly increase the complexity of the model, which made it difficult to apply the model for other purposes. In author's opinion, it is better to add a simplify function in Revit to substitute original model objects with simple configurations. It is similar to the mass function, but this function should be able to create a similar shape of mass of each model object and replace it automatically. It means users only need to choose the instance they want and click the simplify button to achieve model simplification. Although Revit provides different level of details (fine, middle and coarse), this only changes the texture of materials. The function proposed by this study is to change the shape and the structure—using one single configuration to represent one type of objects. Similar to the simplify process for the windows and doors in this study. The author believes this function is extremely helpful to associate with other tools when the model geometry is the major concern, just like CFD simulation.

6.3.2 Improvement for simulation CFD

Material recognition probably is the most practical and approachable function to develop for Autodesk Simulation CFD. This software is not capable of building the geometry model itself. Other 3D CAD tools are used to create the geometry model. As a digital modeling tool, BIM has been increasingly utilized in most construction projects. The information embedded in the building information model is supposed to be helpful for CFD simulation. The material information in the building information model is available for Simulation CFD. It can facilitate the material defining step. But the CFD tool is incapable to recognize these information.

This function should enable Simulation CFD to recognize most regular model materials automatically when importing geometry model from Revit. Only special materials have to be defined manually. In this way, users can skip material defining step and start from setting boundary condition

Specific interface for ACE users is another suggestion though it is difficult to achieve now. During this study, the biggest barrier that impedes the results' accuracy is because the author is familiar with the CFD tool. Autodesk Simulation CFD is a widely-used tool not only for AEC industry. Users without any mechanical background could find it is difficult to put the right settings.

It is helpful to develop a special plug-in module that concentrates on CFD simulation for AEC industry. The interface of this module turns those CFD terminologies to an understandable way, such as supply air flow rate, room design temperature, etc. And there are some default templates for regular HVAC equipment or default scenario for wind environment analysis. Special equipment can be defined manually through product modeler. Since there are no obstacles to import geometry model from Revit, this kind of plug-in will effectively boost the convenience and accuracy of CFD simulation.

CHAPTER VII

CONCLUSIONS AND LIMITATIONS

7.1 Conclusion

This study went through the entire process of using building information model for CFD simulation in different model units. It indicates that although building information model has rich geometry information, small defects and flaws make it problematic to be used as CFD geometry model directly. In order to avoid and fix those problems, models need to be pre-processed. The following points show some details and specific information about the conclusion

a) From the perspective of the software function, the building information model software (Revit) is totally capable of supporting the CFD simulation. But the model defects like gaps on the façade and missing HVAC equipment impede the collaboration process

b) A standardize workflow to process the building information model is necessary. The specific steps can follow the "pre-process" in this research. It not only helps to solve the basic airtight problem, but also facilitates the entire simulation process.

c) There are some problems and barriers during the collaboration of these two tools. Problems like airtightness and lack of element would make the model unable to be simulated; problems like detailed model objects and unnecessary will be harmful for the efficiency of CFD simulation.

d) It is difficult for a personal computer to solve the large model simulation work. Solving the whole building test model eventually failed after a long time of process in this study. Large models usually come with much more detailed data settings. And the irregular interior layout also makes it even worse.

In conclusion, the most fundamental reason for these problems is the different inclinations of these two tools. The building information model is pursuing the authenticity of geometry model with comprehensive and accurate data in it. It requires the model as elaborate as possible. In Simulation CFD, computing and processing is the essential focus. Coarse geometry model is helpful to simplify the calculation. This natural difference in preference has automatically created a gap between them. The wider application scope makes it difficult for CFD to study some specific building indoor cases. Thus, even it is possible to integrate these two tools, extra preparation work is indispensable.

7.2 Limitations

There are many problems and limitations due to the limited time and the hardware problems. The depth and the width of these two subjects are inadequate because of lack of inter-discipline background. Here are some major limitations of this study.

a) Validation step is missing. Validation is supposed to be one of the most important steps to test the accuracy of the CFD simulation. Other similar studies prefer to use equipment such as infrared camera or anemometer to show the indoor temperature distribution and airflow velocity. This study only showed the final indoor airflow visualization without a solid validation.

b) Limitations of software. Apart from the BIM and CFD software in this study, there are many other good BIM software, some large construction companies even have their own programs. And the CFD solution is also supported by many other applications such as DesignBuilder. It is impossible to test all these tools, some of them may have a better and more efficient way to work together. Further verification is required to find out their interoperability.

c) Modelling method comparison. This study only investigated using building information model to create CFD geometry model. A regular way to create the model from the sketch has not been tested in this study. Therefore, there is no way to decide which one will take less time. Also it is uncertain that if using building information model is a quicker way to conduct the CFD simulation. However, this study provided some important values like the time spend on modify building information model. It can be used in future studies.

d) Hardware limitation. The entire building simulation was unsuccessful because the tremendous computing workload crashed the computer. It did not mean CFD is not capable of simulating whole building scale. A better computer may provide a stronger support to this study.

Based on these limitations, the following studies can expand the variety of this topic by combining different BIM & CFD applications and programs to test their interoperability. It is also necessary to simulate and validate different types of HVAC system to confirm their practicability. Moreover, the standard process or algorithm to optimize the building information model for CFD analysis remains to be developed in the future.

REFERENCES

Architecture2030 (2013). "WHY THE BUILDING SECTOR?", <http://architecture2030.org/buildings_problem_why/>. (May, 2015).

Autodesk (2012). "BUILD BETTER HEALTHCARE BUILDINGS WITH BIM AND CFD." <http://static-dc.autodesk.net/content/dam/autodesk/www/solutions/building-information-modeling/building-design/build-better-healthcare-buildings-with-bim-whitepaper.pdf>. (May. 2015).

Cehlin, M. (2006). "Visualization of air flow, temperature and concentration indoors: whole-field measuring methods and CFD."

Chen, Q., and Srebric, J. (2002). "A procedure for verification, validation, and reporting of indoor environment CFD analyses." *HVAC&R Research*, 8(2), 201-216.

Comas, J. T. (2014). "Numerical Simulations of Multiphase Flows." Master, Sorbonne University.

Den Hartog, J., Koutamanis, A., and Luscuere, P. "Possibilities and limitations of CFD simulation for indoor climate analysis." *Proc., Fifth Design and Decision Support Systems in Architecture and Urban Planning-Part one: Architecture Proceedings, Nijkerk, 2000*.

Devlin, N. (2011). "Validation of an active chilled beam design for a healthcare facility." *People*, 190, 133.

Gousseau, P., Blocken, B., and Van Heijst, G. (2011). "CFD simulation of pollutant dispersion around isolated buildings: On the role of convective and turbulent mass fluxes in the prediction accuracy." *Journal of Hazardous Materials*, 194, 422-434.

Ham, Y., and Golparvar-Fard, M. "Identification of Potential Areas for Building Retrofit Using Thermal Digital Imagery and CFD Models." *Proc., Computing in Civil Engineering (2012)*, ASCE, 642-649.

Ivanov, B., Zhelondz, O., Borodulkin, L., and Ruser, H. "Distributed smart sensor system for indoor climate monitoring." *Proc., KONNEX Scientific Conf., Mnchen*, 10-11.

Karlsson, J. F., and Moshfegh, B. (2005). "Investigation of indoor climate and power usage in a data center." *Energy and Buildings*, 37(10), 1075-1083.

Lee, S., and Song, D. "Prediction and evaluation method of wind environment in the early design stage using bim-based cfd simulation." *Proc., IOP Conference Series: Materials Science and Engineering*, IOP Publishing, 012035.

Mustakallio, P., Moilanen, T., Ruponen, M., Kosonen, R., Hagström, K., and Oy, H. "Development of CFD model for a cooled beam–importance of product specific boundary conditions." *Proc., Proceedings of Clima*.

O'Grady, W., and Keane, M. (2006). "Specification of an IFC based software application to support CFD simulation." *ICCCBE 2006*.

Oughton, D., and Hodkinson, S. (2008). *Faber & Kell's Heating & Air-conditioning of Buildings*, Routledge.

Ranade, V. V. (2001). *Computational flow modeling for chemical reactor engineering*, Academic press.

Schauberger, G., Piringer, M., and Petz, E. (2000). "Steady-state balance model to calculate the indoor climate of livestock buildings, demonstrated for finishing pigs." *International Journal of Biometeorology*, 43(4), 154-162.

Sreshthaputra, A., Haberl, J., and Andrews, M. (2004). "Improving building design and operation of a Thai Buddhist temple." *Energy and buildings*, 36(6), 481-494.

Stamou, A., and Katsiris, I. (2006). "Verification of a CFD model for indoor airflow and heat transfer." *Building and Environment*, 41(9), 1171-1181.

Vempati, A. R. (2011). "Computational Fluid Dynamics Investigation of Air Velocity and Temperature Distribution in a Room Equipped with Active Chilled Beam Air-conditioning." University of Florida.

Wargocki, P. (1998). *Human perception, productivity and symptoms related to indoor air quality*, Centre for Indoor Environment and Energy, Department of Energy Engineering, Technical University of Denmark.

Yoon, N., Oshitani, N., and Ando, Y. (2014). "Automated CFD Simulation System With BIM For BCA Green Mark Certification." *ASHARE*, 121.

Zhai, Z. J., and Chen, Q. Y. (2005). "Performance of coupled building energy and CFD simulations." *Energy and buildings*, 37(4), 333-344.